



Model created in COMSOL Multiphysics 6.4

# Kirsch Infinite Plate Problem

## Introduction

---

In this example, you perform a static stress analysis to obtain the stress distribution in the vicinity of a small hole in an infinite plate. Two approximations of the infinite plate are evaluated. The first one uses a plate that is large compared to the hole while the second one employs an infinite element domain.

The problem is a classic benchmark, and the theoretical solution was derived by G. Kirsch in 1898. This implementation is based on the Kirsch plate model described on page 184 in *Mechanics of Materials*, D. Roylance (Ref. 1). The stress level is compared with the theoretical values.

## Model Definition

---

Model the infinite plate in a 2D plane stress approximation as a 2 m-by-2 m plate with a hole with a radius of 0.1 m in the middle. Due to symmetry in load and geometry you need to analyze only a quarter of the plate, see Figure 1. Choose the size of the plate sufficiently large so that the stress concentration close to the hole is not affected.

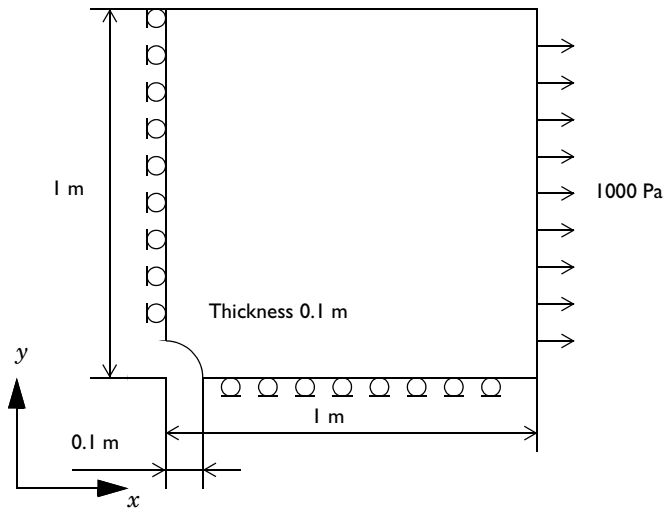


Figure 1: Geometry model of the Kirsch plate with rollers defining the symmetry plane.

When modeling a plate using the infinite element domain you need to create an additional layers around the plate. Those layers simulate the part that stretches to infinity and can have an arbitrarily length along the direction that stretches to infinity, for example 0.1 m.

In the model the infinite element domain is created along the  $y$  direction only since the numerical results along  $x = 0$  symmetry plane are compared to an analytical reference and infinite element domain in  $x$  direction only have a minor influence.

### MATERIAL

Isotropic material with,  $E = 2.1 \cdot 10^{11}$  Pa,  $\nu = 0.3$ .

### LOAD

A distributed stress of  $10^3$  Pa on the right edge pointing in the  $x$  direction.

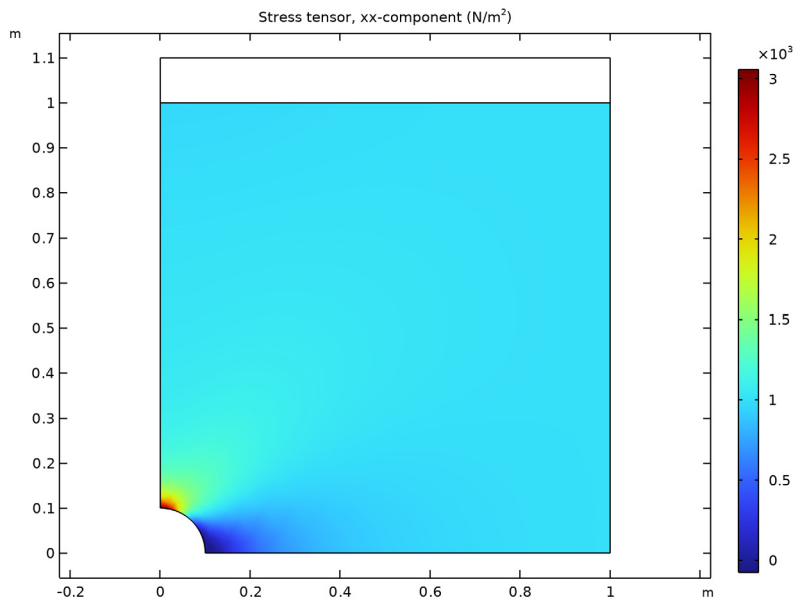
### CONSTRAINTS

Symmetry planes,  $x = 0, y = 0$ .

## Results and Discussion

---

The distribution of the normal stress in the  $x$  direction,  $\sigma_x$ , is shown in [Figure 2](#) and [Figure 3](#). The stress contours of the finite model and the infinite model are very similar.



*Figure 2: Distribution of the normal stress in the  $x$  direction for the finite model.*

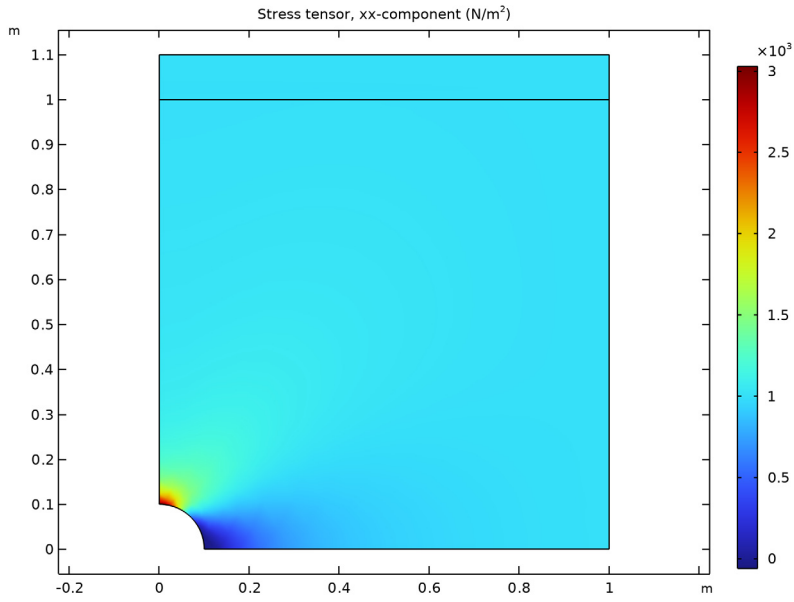


Figure 3: Distribution of the normal stress in the  $x$  direction for the infinite model.

According to Ref. 1 the stress  $\sigma_x$  along the vertical symmetry line can be calculated as

$$\sigma_x = \frac{1000}{2} \left( 2 + \frac{0,1^2}{y^2} + 3 \frac{0,1^4}{y^4} \right) \quad (1)$$

Figure 4 shows the stress  $\sigma_x$  obtained from the solved models, and plotted as a function of the true  $y$ -coordinate along the left symmetry edge, which are in close agreement with the theoretical value according to Equation 1.

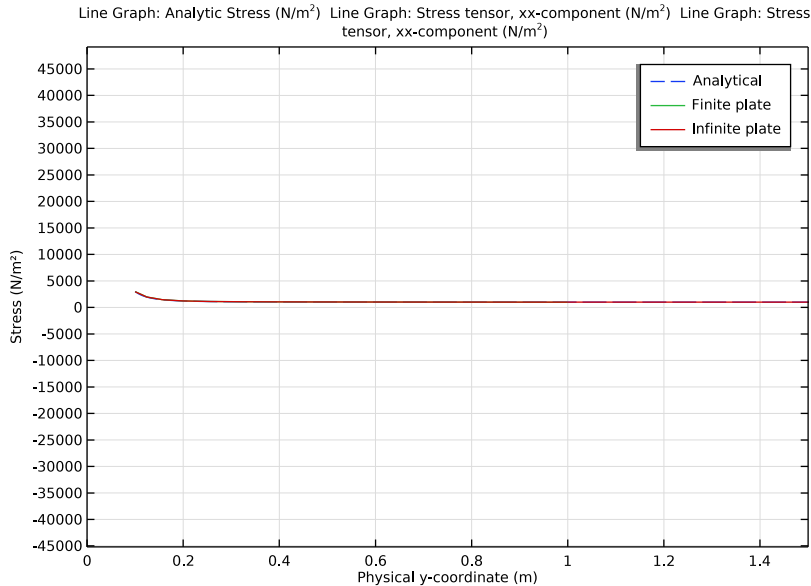


Figure 4: Normal stress, simulated results (solid line) versus the theoretical values (dashed line).

Away from the hole, stresses from the finite model starts drifting from the theoretical values, while stresses from the infinite model matches closely with the theoretical value.

The stress error is summarized in the following table:

TABLE 1: STRESS ERROR RELATIVE TO ANALYTICAL SOLUTION.

	Finite plate	Infinite plate
Near hole	1.1%	0.2%
Away from hole	-4.0%	-0.1%

### Notes About the COMSOL Implementation

The default scaling function in **Infinite Element Domain** is rational. This type of function is well adapted to cases where the degrees of freedom vanish to zero at infinity. The present model is submitted to infinite loads at infinity, that means that constant strain and linear

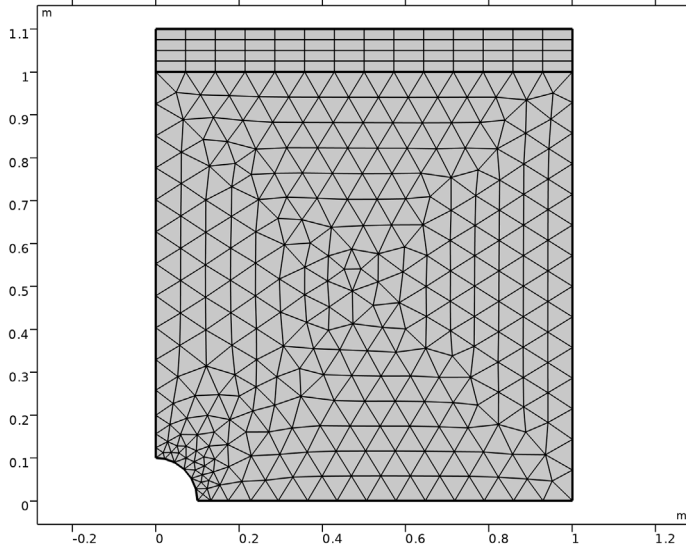
displacement are expected. For this type of infinite solution, polynomial functions are preferred. The relation between the stretched and geometric coordinates is

$$X_m - X_0 = f\left(\frac{X - X_0}{\Delta X}\right)$$

where the function  $f$  is defined by an analytic function. Here we want  $f$  as a second-order polynomial:  $f(\xi) = a\xi^2 + b\xi + c$ . The continuity conditions at  $X_0$ ,  $f(0) = 0$ , and at the end of the domain,  $f(1) = p_w$ , imply that the polynomial is

$$f(\xi) = (p_w - b)\xi^2 + b\xi$$

The infinite element domain gives the best results when meshed with rectangular elements; see [Figure 5](#).



*Figure 5: Infinite element domain modeled with rectangular elements.*

## Reference

1. D. Roylance, *Mechanics of Materials*, John Wiley & Sons, 1996.

---

**Application Library path:** Structural\_Mechanics\_Module/  
Verification\_Examples/kirsch\_plate

---

### *Modeling Instructions*




---

From the **File** menu, choose **New**.

#### **NEW**

In the **New** window, click  **Model Wizard**.

#### **MODEL WIZARD**

- 1 In the **Model Wizard** window, click  **2D**.
- 2 In the **Select Physics** tree, select **Structural Mechanics > Solid Mechanics (solid)**.
- 3 Click **Add**.
- 4 Click  **Study**.
- 5 In the **Select Study** tree, select **General Studies > Stationary**.
- 6 Click  **Done**.

#### **GLOBAL DEFINITIONS**

##### *Parameters 1*


- 1 In the **Model Builder** window, under **Global Definitions** click **Parameters 1**.
- 2 In the **Settings** window for **Parameters**, locate the **Parameters** section.
- 3 In the table, enter the following settings:

<b>Name</b>	<b>Expression</b>	<b>Value</b>	<b>Description</b>
pw	10[m]	10 m	Physical width of infinite element domain
deltaY	0.1[m]	0.1 m	Geometric thickness of infinite element layer

Draw a rectangle with a top layer that represents the infinite element domain.

#### **GEOMETRY 1**



##### *Rectangle 1 (r1)*

- 1 In the **Geometry** toolbar, click  **Rectangle**.



- 2 In the **Settings** window for **Rectangle**, locate the **Size and Shape** section.
- 3 In the **Height** text field, type  $1+\text{deltaY}$ .
- 4 Click to expand the **Layers** section. Clear the **Layers on bottom** checkbox.
- 5 Select the **Layers on top** checkbox.
- 6 In the table, enter the following settings:

Layer name	Thickness (m)
Layer 1	deltaY

#### *Circle 1 (c1)*

- 1 In the **Geometry** toolbar, click  **Circle**.
- 2 In the **Settings** window for **Circle**, locate the **Size and Shape** section.
- 3 In the **Radius** text field, type 0.1.
- 4 Click  **Build Selected**.


#### *Difference 1 (dif1)*

- 1 In the **Geometry** toolbar, click  **Booleans and Partitions** and choose **Difference**.
- 2 Add the rectangle and remove the circle in the **Difference** section.
- 3 In the **Settings** window for **Difference**, click  **Build All Objects**.

### GLOBAL DEFINITIONS

First add an analytical function for stress, based on Kirsch's theoretical solution of an infinite plate.

#### *Analytic Stress*

- 1 In the **Home** toolbar, click  **Functions** and choose **Global > Analytic**.
- 2 In the **Settings** window for **Analytic**, type Analytic Stress in the **Label** text field.
- 3 In the **Function name** text field, type AnaStress.
- 4 Locate the **Definition** section. In the **Expression** text field, type  $1000/2*(2+(0.1/y)^2+3*(0.1/y)^4)$ .
- 5 In the **Arguments** text field, type  $y$ .
- 6 Locate the **Units** section. In the table, enter the following settings:


Argument	Unit
$y$	m

- 7 In the **Function** text field, type  $N/m^2$ .

Create an analytic polynomial function to define the scaling in the infinite element domain.

## DEFINITIONS


### Analytic 2 (an2)

- 1 In the **Definitions** toolbar, click  **Analytic**.
- 2 In the **Settings** window for **Analytic**, locate the **Definition** section.
- 3 In the **Expression** text field, type  $(pw-10*\text{deltaY})*x^2+10*\text{deltaY}*x$ .
- 4 Locate the **Units** section. In the table, enter the following settings:

Argument	Unit
x	m

- 5 In the **Function** text field, type m.

### Infinite Element Domain 1 (ie1)

- 1 In the **Definitions** toolbar, click  **Infinite Element Domain**.
- 2 Select Domain 2 only.
- 3 In the **Settings** window for **Infinite Element Domain**, locate the **Scaling** section.
- 4 From the **Coordinate stretching type** list, choose **User defined**.
- 5 From the **Stretching function** list, choose **Analytic 2 (an2)**.

Add a variable representing the physical y-coordinate to be used in postprocessing.

### Variables 1


- 1 In the **Model Builder** window, under **Component 1 (comp1)** right-click **Definitions** and choose **Variables**.
- 2 In the **Settings** window for **Variables**, locate the **Variables** section.
- 3 In the table, enter the following settings:

Name	Expression	Unit	Description
ym	if(dom==2,ie1.Ym,y)	m	Physical y-coordinate


## SOLID MECHANICS (SOLID)

First set up a model without the **Infinite Element Domain**.


- 1 In the **Model Builder** window, under **Component 1 (comp1)** click **Solid Mechanics (solid)**.
- 2 In the **Settings** window for **Solid Mechanics**, locate the **Domain Selection** section.
- 3 In the list box, select **2 (infinite elements)**.

- 4 Click  **Remove from Selection**.
- 5 Select Domain 1 only.
- 6 Locate the **2D Approximation** section. From the list, choose **Plane stress**.
- 7 Locate the **Thickness** section. In the  $d$  text field, type 0.1.

*Symmetry 1*

- 1 In the **Physics** toolbar, click  **Boundaries** and choose **Symmetry**.
- 2 Select Boundaries 1, 2, and 5 only.



*Boundary Load 1*

- 1 In the **Physics** toolbar, click  **Boundaries** and choose **Boundary Load**.
- 2 Select Boundaries 6 and 7 only.
- 3 In the **Settings** window for **Boundary Load**, locate the **Force** section.
- 4 Specify the  $\mathbf{f}_A$  vector as

1e3	x
0	y

Now set up a model with the infinite element domain.

**ADD PHYSICS**

- 1 In the **Home** toolbar, click  **Add Physics** to open the **Add Physics** window.
- 2 Go to the **Add Physics** window.
- 3 In the tree, select **Structural Mechanics > Solid Mechanics (solid)**.
- 4 Click the **Add to Component 1** button in the window toolbar.
- 5 In the **Home** toolbar, click  **Add Physics** to close the **Add Physics** window.

**SOLID MECHANICS 2 (SOLID2)**

- 1 In the **Settings** window for **Solid Mechanics**, locate the **2D Approximation** section.
- 2 From the list, choose **Plane stress**.
- 3 Locate the **Thickness** section. In the  $d$  text field, type 0.1.

*Symmetry 1*

- 1 In the **Physics** toolbar, click  **Boundaries** and choose **Symmetry**.
- 2 Select Boundaries 1, 2, and 5 only.

*Boundary Load 1*

- 1 In the **Physics** toolbar, click  **Boundaries** and choose **Boundary Load**.

- 2 Select Boundaries 6 and 7 only.
- 3 In the **Settings** window for **Boundary Load**, locate the **Force** section.
- 4 Specify the  $\mathbf{f}_A$  vector as

1e3	x
0	y

## MATERIALS

### Material 1 (mat1)


- 1 In the **Model Builder** window, under **Component 1 (comp1)** right-click **Materials** and choose **Blank Material**.
- 2 In the **Settings** window for **Material**, locate the **Material Contents** section.
- 3 In the table, enter the following settings:

Property	Variable	Value	Unit	Property group
Young's modulus	E	2.1e11	Pa	Young's modulus and Poisson's ratio
Poisson's ratio	nu	0.3	1	Young's modulus and Poisson's ratio
Density	rho	7800	kg/m <sup>3</sup>	Basic

## MESH 1

For the finite plate selection, a customized free triangular mesh must be used for getting a better solution in the stress concentration region. A smaller element size is set at the expected location of stress concentration.

### Free Triangular 1

- 1 In the **Mesh** toolbar, click  **Free Triangular**.
- 2 In the **Settings** window for **Free Triangular**, locate the **Domain Selection** section.
- 3 From the **Geometric entity level** list, choose **Domain**.
- 4 Select Domain 1 only.


### Size 1

- 1 Right-click **Free Triangular 1** and choose **Size**.
- 2 In the **Settings** window for **Size**, locate the **Geometric Entity Selection** section.
- 3 From the **Geometric entity level** list, choose **Point**.
- 4 Select Point 1 only.


- 5 Locate the **Element Size** section. Click the **Custom** button.
- 6 Locate the **Element Size Parameters** section.
- 7 Select the **Maximum element size** checkbox. In the associated text field, type 0.02.

Infinite elements give better results when meshed with rectangular elements.


#### *Mapped 1*

In the **Mesh** toolbar, click  **Mapped**.

#### *Distribution 1*

- 1 Right-click **Mapped 1** and choose **Distribution**.
- 2 Select Boundary 2 only.
- 3 In the **Settings** window for **Distribution**, locate the **Distribution** section.
- 4 In the **Number of elements** text field, type 4.
- 5 Click  **Build All**.


#### **STUDY 1**

In the **Study** toolbar, click  **Compute**.

#### **RESULTS**

To check the error in the computed results, make a point evaluation of stresses near the hole ( $y = 0.1$ ) and away from the hole ( $y = 1$ ) for the solution computed with and without the infinite element domain. The error can be determined by finding the difference between computed stresses and analytical stresses.

#### *Error Evaluation*

- 1 In the **Results** toolbar, click  **Point Evaluation**.
- 2 In the **Settings** window for **Point Evaluation**, type Error Evaluation in the **Label** text field.
- 3 Select Points 1 and 2 only.
- 4 Locate the **Expressions** section. In the table, enter the following settings:

<b>Expression</b>	<b>Unit</b>	<b>Description</b>
$(\text{solid.sx-AnaStress}(y))/\text{AnaStress}(y)$		Error in finite plate
$(\text{solid2.sx-AnaStress}(y))/\text{AnaStress}(y)$		Error in infinite plate

- 5 Click  **Evaluate**.

### *Stress (solid)*

The default plots show the von Mises stress combined with a scaled deformation of the plate. Remove deformation and display the stress field in the  $x$  direction instead since the external load is oriented in that direction.

#### *Surface 1*

- 1 In the **Model Builder** window, expand the **Stress (solid)** node, then click **Surface 1**.
- 2 In the **Settings** window for **Surface**, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Component 1 (comp1) > Solid Mechanics > Stress > Stress tensor (spatial frame) - N/m<sup>2</sup> > solid.sGp<sub>xx</sub> - Stress tensor, xx-component**.
- 3 Locate the **Coloring and Style** section. From the **Color table** list, choose **Rainbow**.

#### *Deformation*

- 1 In the **Model Builder** window, expand the **Surface 1** node.
- 2 Right-click **Deformation** and choose **Delete**.


#### *Surface 1*

- 1 In the **Model Builder** window, expand the **Results > Stress (solid2)** node, then click **Surface 1**.
- 2 In the **Settings** window for **Surface**, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Component 1 (comp1) > Solid Mechanics 2 > Stress > Stress tensor (spatial frame) - N/m<sup>2</sup> > solid2.sGp<sub>xx</sub> - Stress tensor, xx-component**.
- 3 Locate the **Coloring and Style** section. From the **Color table** list, choose **Rainbow**.

#### *Deformation*


- 1 In the **Model Builder** window, expand the **Surface 1** node.
- 2 Right-click **Deformation** and choose **Delete**.

#### *Stress Profile*

- 1 In the **Results** toolbar, click  **ID Plot Group**.
- 2 In the **Settings** window for **ID Plot Group**, type **Stress Profile** in the **Label** text field.

#### *Line Graph 1*

- 1 Right-click **Stress Profile** and choose **Line Graph**.
- 2 Select **Boundaries 1** and **2** only.
- 3 In the **Settings** window for **Line Graph**, locate the **y-Axis Data** section.
- 4 In the **Expression** text field, type **AnaStress (ym)**.
- 5 Locate the **x-Axis Data** section. From the **Parameter** list, choose **Expression**.


- 6 In the **Expression** text field, type  $ym$ .
- 7 In the **Stress Profile** toolbar, click  **Plot**.
- 8 Click to expand the **Legends** section. From the **Legends** list, choose **Manual**.
- 9 Select the **Show legends** checkbox.
- 10 In the table, enter the following settings:

---

**Legends**

---

Analytical

- 11 Click to expand the **Coloring and Style** section. Find the **Line style** subsection. From the **Line** list, choose **Dashed**.
- 12 In the **Stress Profile** toolbar, click  **Plot**.

### *Line Graph 2*

- 1 In the **Model Builder** window, right-click **Stress Profile** and choose **Line Graph**.
- 2 Select Boundary 1 only.
- 3 In the **Settings** window for **Line Graph**, click **Replace Expression** in the upper-right corner of the **y-Axis Data** section. From the menu, choose **Component 1 (comp1) > Solid Mechanics > Stress > Stress tensor (spatial frame) - N/m<sup>2</sup> > solid.sGp<sub>xx</sub> - Stress tensor, xx-component**.
- 4 Locate the **x-Axis Data** section. From the **Parameter** list, choose **Expression**.
- 5 In the **Expression** text field, type  $y$ .
- 6 Locate the **Legends** section. Select the **Show legends** checkbox.
- 7 From the **Legends** list, choose **Manual**.
- 8 In the table, enter the following settings:

---

**Legends**

---

Finite plate

### *Line Graph 3*

- 1 Right-click **Stress Profile** and choose **Line Graph**.
- 2 Select Boundaries 1 and 2 only.
- 3 In the **Settings** window for **Line Graph**, click **Replace Expression** in the upper-right corner of the **y-Axis Data** section. From the menu, choose **Component 1 (comp1) > Solid Mechanics 2 > Stress > Stress tensor (spatial frame) - N/m<sup>2</sup> > solid2.sGp<sub>xx</sub> - Stress tensor, xx-component**.

- 4 Locate the **x-Axis Data** section. From the **Parameter** list, choose **Expression**.
- 5 In the **Expression** text field, type  $ym$ .
- 6 Locate the **Legends** section. Select the **Show legends** checkbox.
- 7 From the **Legends** list, choose **Manual**.
- 8 In the table, enter the following settings:

---


**Legends**

---

Infinite plate

---

*Stress Profile*

- 1 In the **Model Builder** window, click **Stress Profile**.
- 2 In the **Settings** window for **ID Plot Group**, locate the **Axis** section.
- 3 Select the **Manual axis limits** checkbox.
- 4 In the **x minimum** text field, type 0.
- 5 In the **x maximum** text field, type 1.5.
- 6 Locate the **Plot Settings** section.
- 7 Select the **x-axis label** checkbox. In the associated text field, type Physical  $y$ -coordinate (m).
- 8 Select the **y-axis label** checkbox. In the associated text field, type Stress (N/m<sup>2</sup>).
- 9 In the **Stress Profile** toolbar, click  **Plot**.